International Journal of Research in Engineering and Science (IJRES)
ISSN (Online): 2320-9364, ISSN (Print): 2320-9356
www.ijres.org Volume 6 Issue 8 Ver. II ǁ 2018 ǁ PP. 12-17

Fluent-Based Simulation And Analysis On The Performance Of Automotive High Voltage Cooling Fan

Du Wei,Chen Lingshan,Wu Hao
Vehicle Engineering institute,Shanghai University of Engineering Science,Shanghai,201620,China
Corresponding Author :Du Wei

***ABSTRACT**：Based on the computational fluid dynamics method, the high voltage cooling fan is analyzed via modeling by CATIA and simulation by Fluent. By studying the impact of fan structure upon fan performance, the hub ratio, the blade angle and the number of blades are obtained. Finally, the influence of fan structure parameters on fan performance is speculated.therefore, the approach provides a reference to the fan design and optimization.*

***KEY WORDS**: cooling fan; Fluent; performance analysis*

Date of Submission:17-10-2018 Date of acceptance: 03-11-2018

I. INTRODUCTION

Thehigh voltage cooling fan is an important part of the fuel cell vehicle cooling system, and its performance directly affects the function and life of the cells. Therefore,improving high voltage fan performance and efficiency have become important directions for structural optimization of fuel cell vehicle cooling fans.[1]

The traditional fan optimization method is based on a large number of experimental studies, this approach requires long development cycle and high cost. With the continuous development of CFD (Computational Fluid Dynamics)technology and computer hardware, Computational optimization techniques based on 3D RANS turbulence model are widely used. MOREAU[2] used CFD to design and calculate automotive cooling fans in 1997, and verify that the model is valid through experiments.He Qi et al. [3]used CFD software to calculate the flow field of the automobile engine cooling fan, and achieved the initial test of the fan performance. Sun Xiaofeng[4] studied the influence of the unequal distance distribution of blades on the aerodynamic performance of the fan, and obtained the optimal arrangement law of the unequal pitch blades. Wu Min et al. [1]analyzed the internal pressure field and velocity field of the fan and concluded that the tip clearance is an important cause of low fan efficiency. It is suggested that the tip clearance should be reduced or the rotating ring should be installed to improve the fan performance. Dong Xiaobin et al. [5]using Fluent software to study the effects of different airfoil sections on fan performance.

LEE et al. [6]studied the effect of low-speed axial fan blade inclination, blade thickness and the position of maximum blade thickness on fan efficiency by response surface method. The effect of blade inclination on fan efficiency is obvious.

Summarizing the above research results, it can be concluded that the fan structure has a great influence on the performance of the fan. Therefore, it is important to analyze the influence of the fan structure on the performance of the fan structure optimization design. In this paper, through the three-dimensional modeling, the CFD model of fan aerodynamic performance is calculated, the flow field analysis of the fan is carried out, and the influence of fan blade inclination angle, hub ratio and number of blades on the aerodynamic performance of the fan is studied to guide the improved design of the fan.

1.3D model of cooling fan

The basic parameters of fuel cell vehicle cooling fan are as follows , the outer diameter is 304 mm, hub diameter is 102 mm, the number of blades is 7, and the blade inclination is 35°.The three-dimensional simplified model is shown in Figure 1.

Fig.1 simplified 3D model of cooling fan

II. BASIC PRINCIPLES OF COMPUTATIONAL FLUID DYNAMICS

2.1 fundamental equation

Fluid flow is subject to the law of conservation of physics, and it is necessary to satisfy the law of conservation of mass, the law of conservation of momentum and the law of conservation of energy.Since the air Mach number flowing through the cooling fan is less than 0.3, air can be treated as an non-compressible fluid.When analyzing and calculating, the heat generated by friction in the flow is very small, so energy conservation is not considered.

Mass conservation equation:

$$\frac{\partial \mathrm{u}}{\partial x}+\frac{\partial v}{\partial \mathrm{y}}+\frac{\partial w}{\partial z}=0$$

In the formula: u, v, w are the flow rates of air in the x, y, and z directions of the coordinate axes ,respectively.

Momentum conservation equation:

$$\frac{\partial(\rho u)}{\partial t}+\mathrm{div}(\rho uu)=\mathrm{div}(\mu \mathrm{grad}\, u)-\frac{\partial p}{\partial x}+Su$$

$$\frac{\partial(\rho v)}{\partial t}+\mathrm{div}(\rho vu)=\mathrm{div}(\mu \mathrm{grad}\, v)-\frac{\partial p}{\partial y}+Sv$$

$$\frac{\partial(\rho w)}{\partial t}+\mathrm{div}(\rho wu)=\mathrm{div}(\mu \mathrm{grad}\, w)-\frac{\partial p}{\partial z}+Sw$$

In the formula:ρis the air density;μ is the dynamic viscosity;p is the pressure on the fluid micro-body;t is time; Su, Sv and Sw are generalized source terms of momentum conservation equations.For non-compressible fluids with constant viscosity,Su, Sv and Sw are the forces in the x, y and z directions of the micro-element, respectively.

2.2 Turbulence model

The fan flow field is calculated with the RNG k-ε turbulence model,It can better handle high flow rate and flow curve bending by correcting the turbulent viscosity and considering the rotation and swirl flow in the average flow.Turbulence model is as follows:

$$\frac{\partial(\rho\kappa)}{\partial t}+\frac{\partial(\rho\kappa u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[a_\kappa\mu_e\frac{\partial\kappa}{\partial x_j}\right]+G_\kappa+\rho\varepsilon$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho\varepsilon u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[a_\varepsilon\mu_e\frac{\partial\varepsilon}{\partial x_j}\right]+\frac{C_{1\varepsilon}^*\varepsilon}{\kappa}G_\kappa-C_{2\varepsilon}\rho\frac{\varepsilon^2}{\kappa}$$

In the formula:κ, ε are the kinetic energy and dissipation rate, respectively;G_κ is the generation term of the kinetic energy κ caused by the average velocity gradient,

$$G_\kappa=\mu_t\cdot\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_j}{\partial x_j};\mu_e=\mu+\mu_t;\mu_t=\rho C_\mu\frac{\kappa^2}{\varepsilon};C_\mu=0.0845;$$

$$a_\kappa=a_\varepsilon=1.39;C_{1\varepsilon}^*=C_{1\varepsilon}-\frac{\eta(1-\frac{\eta}{\eta_0})}{1+\rho\eta^\varepsilon};C_{1\varepsilon}=1.42;C_{2\varepsilon}=1.68;$$

$$\eta=(2E_{ij}\bullet E_{ij})^{1/2\frac{\kappa}{\varepsilon}};E_{ij}=\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right);\eta_0=4.377;\beta=0.012.$$

III. COOLING FAN SIMULATION MODEL

3.1. Establishment of cooling fan simulation model

Import fan model into Fluent pre-processing software ICEM,establish CFD simulation model according to fan performance test requirements.The model is divided into four parts: the inlet zone, the exit zone, the rotating fluid zone and the pipeline zone.Considering the variation of the flow field in different regions of the engine cooling fan simulation model, the method of partitioning the grid is adopted.The mesh size of the rotating fluid zone grid is small, the mesh size of the pipe zone is slightly larger, and the mesh of the inlet zone and the exit zone is the largest.The model after meshing is shown in Figure 2.

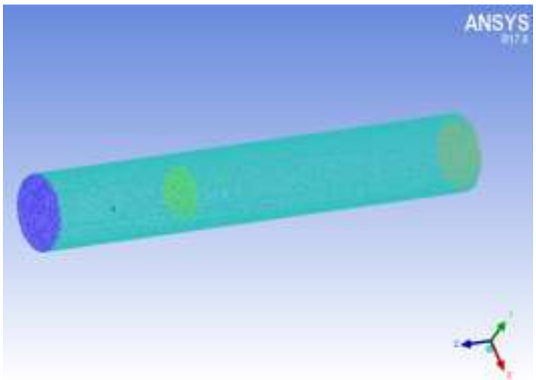

Fig.2 CFD simulation model of cooling fan

3.2. Boundary condition setting

The boundary conditions of the fan flow area mainly include wall boundary conditions, inlet boundary conditions, and exit boundary conditions.In addition, the rotating fluid zone is the only "moving" zone in the four zones, the zone type is defined as the fluid.Import and outlet are set as pressure inlet and pressure outlet respectively.The total flow pressure at the inlet is atmospheric pressure, and the pressure at the outlet is defined as zero relative to atmospheric pressure.Set the fan speed with the rotating coordinate system is 2700 rpm.

IV. SIMULATION RESULT ANALYSIS

4.1. Overall performance analysis of cooling fan

The air flow diagram of the flow area is shown in Figure 3.The airflow is substantially laminar in the inlet zone, after the action of the fan, the airflow speed changes and spirals out to the outlet.The cooling fan surface speed vector diagram shown in Figure 4.It can be seen from Figure 4 that the airflow passes through the fan, the flow direction changes, the rotation occurs with the blade, and a large rotation speed is obtained at the tip of the blade.

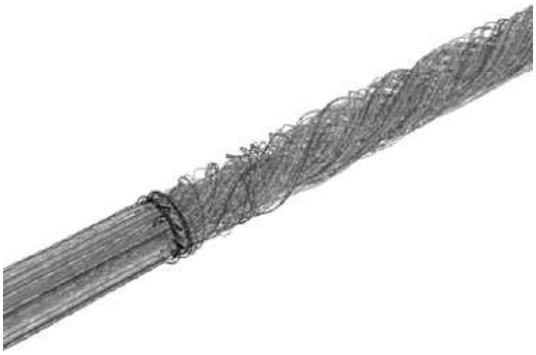

Fig.3 Air stream of flow field

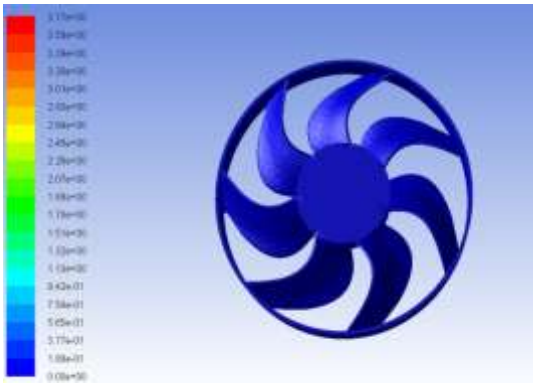

Fig.4 Velocity vector on the surface of cooling fan

A pressure distribution diagram of the Y=0 cross section is shown in Figure 5. The airflow is unevenly distributed after passing through the fan，The diameter of the outlet zone is close to the diameter of the fan, and the pressure at other diameters is small.As the airflow flows toward the exit, the pressure in each area gradually becomes uniform.At the same time, it can be seen from Figure 5 that the outlet zone forms a negative pressure zone along the axis due to insufficient air dynamic pressure.The path-lines of air through cooling fan is shown in Figure 6. As can be seen from Figure 6, the gas in the negative pressure zone produces a back-flow, which will result in a decrease in fan efficiency.

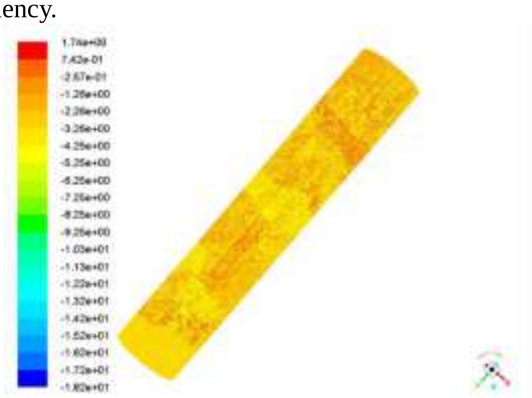

Fig.5 Y=0 sectional pressure contours

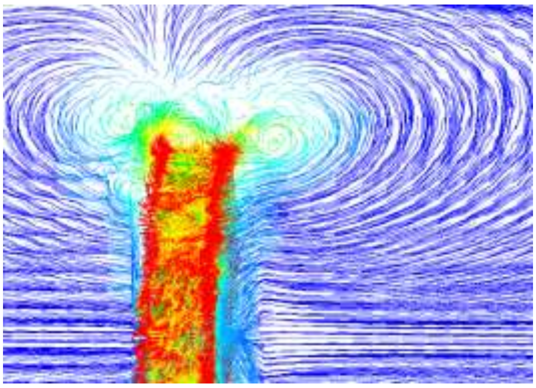

Fig.6 the path-lines of air through cooling fan

4.2. Effect of fan structure parameters on fan performance

According to the design requirements of the axial flow fan and summarize the previous analysis results of the structure of the automotive cooling fan,select the appropriate fan structure parameter variation range, as shown in Table 1.Change the fan structure parameters, calculate and analyze the influence of the fan on the performance.

Tab.1 Fan structure parameter value interval

Fan hub ratio	Fan blade angle/(˚)	blades number
0.29 ~ 0.37	25 ~ 45	5 ~ 9

(1) Effect of fan hub ratio.
Change the fan hub ratio and analyze its effect on fan flow and total pressure efficiency, as shown in Figure 7.

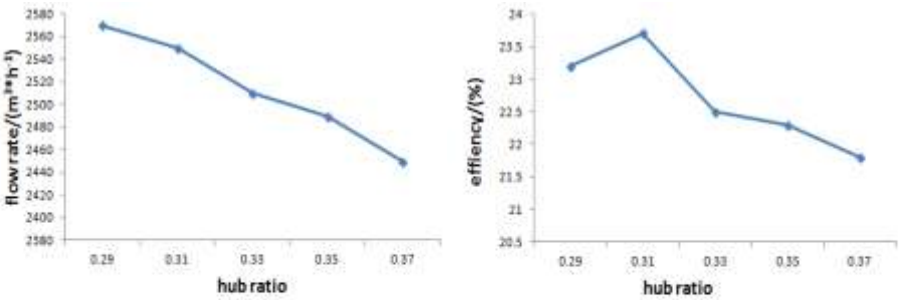

Fig.7 Influence of hub ratio

Analysis of Figure 7 shows that in the interval of 0.29 to 0.37, as the hub ratio increases, the fan flow tends to decrease, and the overall efficiency shows a downward trend after a upward trend first.As the fan hub ratio increases, the effective gas flow area is reduced, and the gas mass passing through the fan per unit time is also reduced when the fan speed is constant.If the hub ratio is too small, the boundary layer of the blade root will be separated, which will reduce the fan pressure and affect the fan efficiency[8].
(2) Effect of fan blade angle.
Change the angle of the fan blade and analyze its effect on fan flow and total pressure efficiency, as shown in Figure 8.

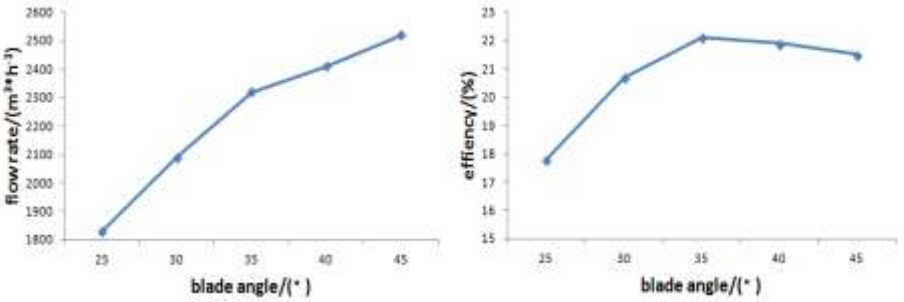

Fig.8 Influence of blade angle

It can be seen from Figure 8 that in the range of 25°to 45°, the fan flow rate increases with the increase of the blade inclination angle, and the increase rate gradually becomes smaller.The efficiency increases first and then decreases with the increase of the blade inclination angle, and the maximum value is obtained near 35°.
(3) Effect of fan blades number
Change the number of blades of the fan and analyze its effect on fan flow and total pressure efficiency, as shown in Figure 9.

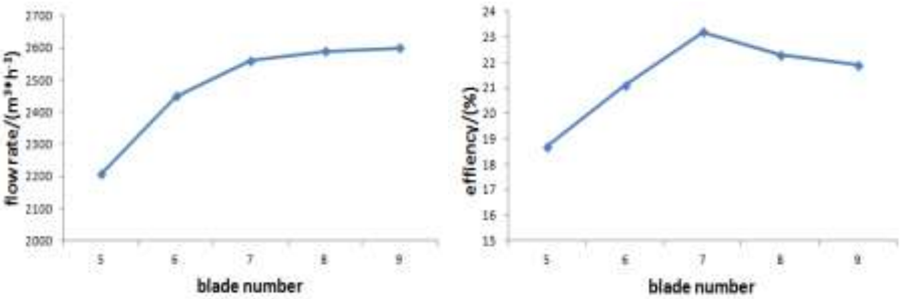

Fig.9 Influence of the number of blades

It can be seen from Figure 9 that in the interval of 5-9, the fan flow increases with the increase of the number of blades.This is because the increase in the number of blades increases the area of action of the blades on the air, thereby increasing the air flow.The fan efficiency increases first and then decreases with the increase of the number of blades.This is because the excessive number of blades will lead to an increase in the equivalent diffusion angle of the blade flow path, thereby increasing the additional loss when the kinetic energy is converted into static pressure energy, which causes the fan pressure and efficiency to decrease.

V. CONCLUSION

In this paper, through fluid dynamics calculation and analysis, the flow field distribution inside the air duct is obtained, and the aerodynamic performance of the fan is evaluated.The influence of fan structure on fan flow and efficiency is analyzed, and the following conclusions are drawn:

(1) The computational fluid dynamics method is used to simulate the performance of the fan, and the distribution law of the internal flow field of the air duct is obtained. The cost is low and the efficiency is high, which has good analytical application value.

(2) The flow field analysis found that there is a negative pressure zone in the exit zone of the air duct and caused gas back-flow, which will affect the efficiency and vibration noise of the fan.

(3) Change the fan hub ratio, blade pitch and number of blades to determine the effect of different parameter values on fan flow and efficiency.The results show:At the test point, the fan flow rate is larger when the hub ratio is 0.28, the blade inclination is 45°, and the number of blades is 9;The fan efficiency is larger when the hub ratio is 0.3, the blade inclination is 35°, and the number of blades is 7.Fan structure performance optimization needs to consider both flow and efficiency, and select the most suitable structural parameters.

REFERENCES

[1]. Wu Min, Wang Yiyou, Shangguan Wenbin, et al. Calculation method of aerodynamic performance of engine cooling fan [J]. Automotive Technology, 2009 (supplied): 8-11.

[2]. MOREAU S, BENNETT E. Improvement of fan design using CFD[EB/OL]. [2013 - 12 - 26]. http://papers. sae. org/970934/.

[3]. He Qi, Mao Jianguo, He Xiaoming. CFD analysis of the performance of automotive engine cooling fans [J]. Automotive Technology, 2009 (1): 46-48.

[4]. Sun Xiaofeng. Research on aeroacoustic characteristics of unequal blade fans [J]. Journal of Beijing Institute of Aeronautics, 1986 (4): 137-145.

[5]. Dong Xiaobin, Yuan Zhaocheng, Lu Bingwu, et al. Research on high-flow and low-noise cooling fans for heavy-duty trucks [J]. Automotive Technology, 2009 (4): 11-14.

[6]. LEE K S, KIM K Y, SAMAD A. Design optimization of low-speed axial flow fan blade with three-dimensional RANS analysis [J]. Journal of Mechanical Science and Technology, 2008, 22(10): 1864-1869.

[7]. Wang Fujun. Computational Fluid Dynamics Analysis—CFD Software Principles and Applications [M]. Beijing: Tsinghua University Press, 2004.

[8]. Changze Zhou. Axial flow ventilation machine practical technology [M]. Beijing: Mechanical Industry Press, 2005.

Du Wei "Fluent-Based Simulation And Analysis On The Performance Of Automotive High Voltage Cooling Fan "International Journal of Research in Engineering and Science (IJRES), vol. 06, no. 08, 2018, pp. 12-17